OPTIMIZATION OF MOLDING PARAMETER EFFECT TO WARPAGE AND SHRINKAGE BASED ON PLASTIC FLOW SIMULATION SOFTWARE: SWIMMING GOGGLE

LAI SI YING

Thesis submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Manufacturing Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Signature

Name of Supervisor : ENCIK MOHAMED REZA ZALANI MOHAMED SUFFIAN

Position : LECTURER

FACULTY OF MECHANICAL ENGINEERING

UNIVERSITY MALAYSIA PAHANG

Date :

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature

Name : LAI SI YING

ID Number : ME07059

Date : 01 DECEMBER 2010

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Mohamed Reza Zalani Mohamed Suffian for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a PhD program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career.

My sincere thanks go to all my course mates and member of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I am also grateful to my siblings for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my ability to attain my goals.

ABSTRACT

This thesis present about the optimization of molding parameter effect to warpage and shrinkage based on plastic flow simulation software: swimming goggles. The objective of this project is to study, analyze, and optimize the parameter that effect to warpage and shrinkage with using plastic flow simulation software. This project is using 3D scanner and draw swimming goggles in SolidWorks 2007. Swimming goggles lens is made by polycarbonate (PC) while swimming goggles frame is made by Acrylonitrile Butadiene Styrene (ABS). The injection molding parameters are optimized by using MoldFlow Plastics Insight (MPI) 5.0 software in order to get the best parameters with minimum warpage and shrinkage. The four injection molding parameter use is melt temperature, mold temperature, injection pressure and flow rate. The small changes of molding parameters will give higher impact to the plastic characteristics. The comparison between the types of analysis will be made. The comparison is between the deflections, fill times, and volumetric shrinkage. As the conclusion, the changes of injection molding parameters will affect the warpage of the swimming goggles.

ABSTRAK

Tesis ini membentangkan tentang kesan pengoptimuman parameter acuan kepada kelengkunagn dan pengecutan berdasarkan perisian simulasi pengaliran plastik: kacamata berenang. Objektif projeck ini adalah mengaji, menganalisis dan mengoptimumkan parameter yang membawa kesan kepada kelengkungan dan pengecutan dengan menggunakan perisian simulasi pengaliran plastik. Projek ini menggunakan 3 Dimensi alat pengimbas dan melukis kacamata berenang dalam "SolidWorks 2007". Kanta kacamata berenang diperbuat daripada Polikarbonat (PC) manakala rangka kacamata berenang diperbuat daripada Acrylonitrile Butadiene Styrene (ABS). Parameter suntikan acuan dioptimumkan dengan menggunakan perisian "MoldFlow Plastics Insight" (MPI). 5.0 untuk mendapatkan parameter terbaik dengan kelengkungan dan pengecutan yang minimum. Empat parameter suntikan acuan yang digunakan adalah suhu mencair, suhu cetakan, tekanan suntikan dan kelajuan aliran.Perubahan kecil dalam parameter acuan akan memberi kesan besar kepada ciri-ciri plastik.Perbandingan antara jenis-jenis kajian dibuat.Penbandingan itu adalah antara defleksi, masa pengisian, dan isipadu pengecutan. Kesimpulannya, perubahan parameter suntikan acuan memberi kesan kepada kelengkungan dan pengecutan kacamata berenang.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

^{o}C	Degree Celsius
%	Percent
MPa	Mega Pascal
mm	Millimeter
cm^3/s	Centimeter cubic per second

LIST OF ABBREVIATIONS

ABS	Acrylonitrile Butadiene Styrene
MPI	Moldflow Plastic Insight 5.0 Software
MPA	Moldflow Plastic Advisor Software
PC	Polycarbonate

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays, mostly people are depending on plastic products. Hence plastic products play an important role in human daily life. The plastic products are used widely in daily life and industries. Their appearances have made human life easier and simple.

The purpose of the project is to find the optimization of molding parameter effect to warpage and shrinkage of swimming goggle based on plastic flow simulation software. Optimization is the use of specific techniques to determine the most cost effective and efficient solution to a problem or design for a process. This project is study the injection molding product by using plastic flow simulation software. Injection molding product has been used in the daily application such as for household appliances, industry field and sport equipments because plastic materials has low cost and light in weight compare to other material.

As a student of mechanical engineering of University Malaysia Pahang (UMP), student can search and study the field of manufacturing engineering through this project.

1.2 BACKGROUND OF PROJECT

The process for injection molding must be understood clearly. Injection molding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. During the process, molten plastic is injected at high pressure into a mold, which is the inverse of the product's shape. Thermoplastic or thermosetting plastic materials are heat until melts then forced it into a steel mould. It cools down and solidifies before mold part is ejected.

It is an important issue to predict the warpage and shrinkage problem in product for plastic injection molding before the manufacturing process of the product. Warpage is an inconsistent alteration in internal stresses causing deformation or distortion of the material after molding or other fabrication. The warpage will be predicted from the temperature distribution, shrinkage, flow-induced shear stress and other mechanical properties. While shrinkage is the contraction for a plastics casting upon polymerizing and contraction of a molded material such as metal or resin during cooling (Gabriele et al.,2005). .

Parameter is define as a set of facts or a fixed limit which establishes or limits how something can or must happen or be done .During this project, the molding parameter such as injection pressure, mold temperature, melt temperature and flow rate are focus in the injection molding. While warpage and shrinkage will be effect by molding parameters such as back pressure, mold temperature, mold cooling, melt temperature, process conditions, injection pressure, part and mold designs. The ability to optimize a process is depends on the ability of controlling the process with reliable and valid measurements. Any changes in these molding parameters can influence the result for plastic characteristics. Hence, all parameter must be measure accurately to get appropriate result. This will help to find out the best parameter for the injection molding product with minimum warpage and shrinkage (Fausto and Rushmeir, 2002).

Reverse engineering is a process to discover the technological principles of a device object or system through analysis of structure function and operation by using 3D scanning technologies such as structured light digitizers, coordinate measuring machines (CMMs), laser scanners and others then create a 3D virtual model for it. It can use to analyze the process of a product working, components consists in a product, and estimate the cost for produce it (Mathew, 2010).

1.3 OBJECTIVE

1. Study the molding parameters for injection molding of swimming goggle
2. Analyze the parameter effect in injection molding to warpage and shrinkage of swimming goggle
3. Optimize the molding parameter and reduce warpage and shrinkage of swimming goggle with the use of plastic flow simulation software

1.4 PROJECT SCOPE

A lot of research and collection of information related to the title must be prepared before the starting of this project. Those research sources are come from books, articles, journals, and other related material. This process also is under guidance of supervisor to make sure those information is connect with the title of project. Latest knowledge is always apply in project until the project is complete.

Reverse engineering requires extraction of information about an instance of a device object sufficient to replicate that part using appropriate manufacturing techniques. This project is using 3D scanner as reverse engineering to study the injection molding product in Faculty of Mechanical Laboratory in Pekan Campus.

The type of 3D scanner using is CIMCORE INFINITE 2.0.It is a device which can analyzes a real world object and collect data on the object's shape. After that, the data will be used to construct digital and three dimensional (3D) models which is useful for a wide variety of applications. It will moving the arm that the probe will mounted on very slow for many times on the object.3D scanner can create a point cloud of geometric samples on the surface of the object been scanned and those points will extrapolate the shape of the object during reconstruction.

After taking shape of the swimming goggle, those data will be transfer into CAD Software. SolidWorks is using to get all dimension of the product and then imported to the Plastic flow simulation software. Through the plastic flow software, molding parameter can be analyzed. Plastic flow simulation software can optimizes the gate locations and find out all the defect part in the injection mold product through flow analysis. When the software finishes analysis for parameter such as mold temperatures, it will suggest suitable dimensions for balanced feed system automatically. This help to improve injection molding cycle to design better quality product.

The use for mold flow software is to optimize the molding parameter and minimize the warpage and shrinkage to get the best product. Mold flow software uses the data from plastic flow software and help to build better plastic parts with effective time and low cost. The design for product can be optimize by modifying circuit and coolant processing parameters.

Hence, the molding can produce better product with new molding parameter which is optimize with using mechanical method to reduce the warpage and shrinkage for it.

1.5 PROBLEM STATEMENT

The plastic flow simulation software is very important since it provide all the data needed to optimize the molding parameter effect to the warpage and shrinkage of injection mold product. When the best parameter is applied in the manufacturing industry, this will help to produce injection mold product which have high quality and quantity and will not influence the safety and health for the user for that product.

Nowadays, the design of injection mold product becomes more complicated and the requirement for quality characteristic is keep increase. Hence the present method used that is trial and error method is not enough to solve the problem in injection molding industry. This project make a research in plastic flow software and able to find out optimum parameter using in the injection mold product which will increase productivity with lower cost using in manufacturing industry.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In the simplest form, the injection molding process is like the operation of a hypodermic needle, A barrel contain heated plastic that is injected into a close mold that contains a machined, reverse image of the desired product. The primary advantage of this process is that many function and features can be incorporated into the product design. This process will minimize, or eliminate, the amount of secondary work required to produce the same product in other ways or using other materials.

The plastic product begins life in the mold. A mold is constructed using a series of components, including various plates, pins, bushings, pillars, ejector systems, and a multitude of other item used for many purposes.

The mold components that form the molded part are the A and B plates .The shape of the final molded product is determined by the shape of the cavity image that is machined into the A and B plate.

The injection molding product can be control by adjusting and optimize the molding parameter in time, pressure and temperature.

2.2 DEFINITION OF PROCESS OPTIMIZATION

According to Stuart et al. (1997), process optimization is the practice of making changes of adjustments to a process to get results. Optimization is the use of specific techniques to determine the most cost effective and efficient solution to a problem or design for a process. A process is typically a set of equipment arranged, controlled, and operated in a particular way, to produce a product. The product must meet certain specifications, such as a certain production rate, product quality and cost. When we talk about optimizing a process, we are usually trying to maximize one or more of the process specifications, while keeping all others within their range. The design and operation of systems or processes to make them as good as possible in some defined sense. The approaches to optimizing systems are varied and depend on the type of system involved, but the goal of all optimization procedures is to obtain the best result possible (again, in some defined sense) subject to restrictions or constraints that are imposed. While a system may be optimized by treating the system itself, by adjusting various parameters of the process in an effort to obtain better results, it generally is more economical to develop a model of the process and to analyze performance changes that result from adjustments in the model (Stuart et al., 1997).

A key step in the formulation of any optimization problem is the assignment of performance measures (also called performance indices, cost functions, return functions, criterion functions, and performance objectives) that are to be optimized. The success of any optimization result is critically dependent on the selection of meaningful performance measures. In many cases, the actual computational solution approach is secondary. Ways in which multiple performance measures can be incorporated in the optimization process are varied.

2.3 INJECTION MOLDING

Injection molding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mold which is the inverse of the product's shape. After a product is designed by an Industrial Designer or an Engineer, molds are made by a mold maker (or toolmaker) from metal, usually either steel or aluminum, and precision machined to form the features of the desired part. Injection molding is widely use for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of productions, with some commonly made items including bottle caps and outdoor furniture.

Dominic (2008) pointed out that the most commonly used thermoplastic materials are polystyrene (low cost, lacking the strength and longevity of other materials), ABS or acrylonitrile butadiene styrene (a co-polymer or mixture of compounds used for everything from Lego parts to electronics housings) ,nylon (chemically resistant, heat resistant, tough and flexible-used for combs),polypropylene (tough and flexible-used for containers), polyethylene and polyvinyl chloride or PVC (more common in extrusions as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of plasticizer) (Dominic,2008).

Injection molding can also be used to manufacture parts from aluminum of brass. The melting points of these metals are much higher than those of plastics; this make for substantially shorter mold lifetimes despite the use of specialized steels .Nonetheless, the costs compare quite favorably to sand casting, particularly for smaller parts.

Mould making is an important supporting industry because their related products represent more than 70% among the components in consumer products. The high demand for shorter design and manufacturing lead times, good dimensionality and overall quality, and rapid design changes has become the bottlenecks in mould industries. It is a complicated process, and required skilled and experienced mould maker.

Injection molding machines, also known as presses, hold the mold in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can generate. This pressure keeps the mold closed during the injection process. Tonnage can vary from less than 5 tones to 6000 tons, with the higher figures used in comparatively few manufacturing operations (Dominic,2008).

According to definition by Hugh (2001) , the basic process in injection mold is heat a thermoplastic material until it melts and forces it into a hollow (cooled) cavity under pressure to fill the mold. After cooling down, the finished part is removed (Hugh, 2001).

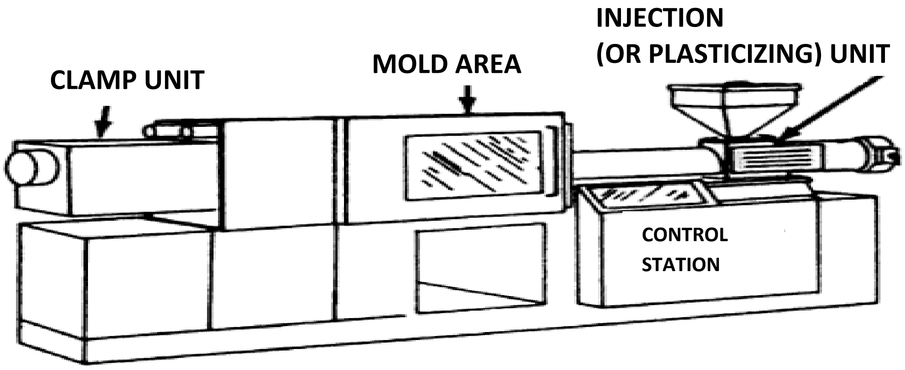

Figure 2.1: General layout for an injection molding molding machine.

Source: Douglas (1999)

A typical injection molding machine shown Figure 2.1 with the covers removed. Plastic pellets are poured in the hopper, and finished parts emerge from the molds.

Injection systems:

1. A material hopper acts as an input buffer
2. A heated chamber (heater bands) melts the material
3. An injector forces the now viscous fluid into the mold

2.4 DEFECTS ON PLASTIC INJECTION MOLDING

The defects such as burn marks or air burn which is brown or black burnt areas on the part located at furthest points from the gate is because of the tool lacks venting or the injection speed is too high. The other type of defect is flash or burrs can be detected on the part when excess material in thin layer exceeding normal part geometry resulting from too much injection speed or too much material injected, clamping force too low or too damaged. Sink marks can be detected as localized depression which happened at thicker zone of product. This defect occurred when the holding time or the pressure too low, cooling time too low with sprueless hot runners and this defect also can be caused by the gate temperature being set too high. The other type of defect is short shot where the finished product is only partial of the original shape. This is because lack of material, injection speed or pressure too low. Warping or shrinkage also defined as twisting is occur when the part is distorted due to cool time is too short, material is too hot, lack of cooling around the tool or incorrect water temperatures (the parts bow inwards towards the cool side of the tool).Weld line or meld line is detected as discolored line where two flow fronts meet. The defect because of the mold or material temperatures set too low which mean the material is cold when they meet so they do not bond uniformly.

2.5 INJECTION MOLDING PARAMETER

When consider the parameter involved with time, temperature, and pressure related factors, it becomes east to pinpoint a parameter that could cause a defect that is not listed in injection molding problem.

For time related factors are cure time, rate of injection, mold open time, time of air blowing, residence time in the cylinder, duration of high and holding pressure application.

For temperature related factors are mold temperature, cylinder temperature, frictional heat from gate and other passages that interfere with the smooth flow of the material, surrounding atmosphere and its effect on mold heat, roughness of runners, sprue diameter, back pressure, screw rpm, nozzle temperature, frictional heat from rate of injection.

For pressure related factor are injection high and injection hold pressure, clamp pressure, back pressure, vents, pressure drops up to gate, cavity pressure.

2.5.1 Time

a) Injection Time

Injection time is the amount of time required for injection depends on how much material is being injected, the viscosity of the material, and the percentage of the machine’s barrel capacity being utilized. The total amount of injection time required is dived into two separate phases which are initial injection and holding injection.

Douglas (1999) examined out that initial injection time is performed using the highest practical pressure for the specific application (normally 10,000-15,000psi [69-103MPa]) in the fastest practical amount of time. When the mold closes completely, either a limit switch or pressure buildup signals the injection screw to push forward and inject the molten plastic into the closed mold. The screw does not turn at this point, but acts only as a plunger to force the material into mold. In most cases, this is less than 2 seconds and rarely more than 3 seconds. Sometimes, depending on machine design, this initial action is divided into two or three smaller actions, or stages. In those cases, the total amount of injection time normally does not exceed 4 to 5 seconds. The initial injection time is controlled through use of a timer. If a “booster” injection time is available, it will be include in the first stage of the initial injection time. When a booster phase is utilized, the injection machine’s entire hydraulic system (injection and clamp) is combined to push a large volume of oil through the system .This can greatly increase the speed at which the material is injected into the mold.

On most machines, the timer for initial injection time (also called injection forward time) controls the total amount of time that the injection screw is pushing forward. The initial injection time is the first part of that time and injection hold is the latter part, on some machines, the hold time and initial time are on separate timers.

The hold time is the amount of time that the injection screw maintains pressure against the plastic after it has been injected into the mold .This pressure is applied against the “cushion” or “pad” and is applied long enough for the gate to freeze off (solidify).When the plastic is injected into the mold, the molten plastic enter the mold cavity image through a gate, This gate is the first point at which the plastic moves into the cavity image. After all the required material goes through the gate and packs the cavity image, the plastic is allowed to cool under hold pressure until it solidifies. But, because it is normally the thinnest part of the cavity image, the gate is the first portion to solidify. At this point, pressure is not applied because the plastic in the cavity lies beyond the solidified gate and the pressure from the injection unit no longer has any effect on it. So the length of time required to hold pressure against the gate is only long